

# Ship-current interactions with TELEMAC

Juliette Parisi, Michael Turnbull, Alan Cooper, James Clarke

HR Wallingford  
Howbery Park, Wallingford, UK  
[j.parisi@hrwallingford.com](mailto:j.parisi@hrwallingford.com)  
[m.turnbull@hrwallingford.com](mailto:m.turnbull@hrwallingford.com)  
[a.cooper@hrwallingford.com](mailto:a.cooper@hrwallingford.com)  
[j.clarke@hrwallingford.com](mailto:j.clarke@hrwallingford.com)

**Abstract** — The Navigation Simulator is often used to train pilots and test harbour or channel designs. It combines hydraulic modelling with ship manoeuvring models and a team of experts (pilots, tug masters, naval architects and navigation specialists), all focused through a suite of real-time simulators.

Currently, the hydrodynamic modelling is an input of the ship simulator, i.e. it is carried out beforehand, the flow fields are then used by the ship simulator. This study presents how the effects of ships have been considered in the hydrodynamic modelling in order to improve ship navigation simulations. Several hydrodynamic scenarios were investigated:

- Representing the ships in the hydrodynamic model by imposing the ship hulls as a pressure field;
- Live communication between the hydrodynamic model and the ship simulator. The position and orientation of the ship is controlled by the ship simulator and passed on to the hydrodynamic model. The resulting flow fields and water elevation are calculated by the hydrodynamic model and fed back to the ship simulator;
- Including propeller wash.

The new developments were used in recent real-time navigation simulations. Feedback from pilots and tug masters is positive. Some instabilities in the hydrodynamic model remain and further developments are underway.

## I. INTRODUCTION

The TELEMAC system flow models, TELEMAC-2D and TELEMAC-3D, are free surface flow models which are not expected to apply to cases where the surface is not open to the atmosphere, including flows under ships and other floating objects. Nevertheless as engineers we frequently need to be able to take account of these things as the flow in a harbour, for example, may be significantly affected by the presence of a floating object and even more by a large moving ship.

Requests to investigate flows in such cases have motivated the following study. While it would seem that conventional CFD simulations are ideally suited to modelling free surface flows including stationary and moving ships because they can model non free surface flows, it would be necessary to set up a two way coupling of the CFD model and TELEMAC, and the CFD computations are likely to be computationally intensive and therefore take too much time in the context of a navigation simulation and would be prohibitive in the context of dynamic linking to a ship simulator.

## II. INFLUENCE OF LARGE SHIPS: CURRENT SHELTERING

TELEMAC-2D and TELEMAC-3D can be used to model ships by imposing a pressure field on the water surface, which represents the ship's hull. The pressure applied is proportional to the depth of the ship in the water. The pressure is given by the equation (1)

$$P = \rho g d \quad (1)$$

- $P$  is the pressure (Pa)
- $\rho$  is the water density ( $\text{kg/m}^3$ )
- $g$  is the acceleration due to gravity ( $\text{m/s}^2$ )
- $d$  is the depth of the ship in the water (m)

### A. Caisson test case

This method was first validated with the caisson test case, against preliminary CFD work carried out by HR Wallingford in 2013. The test case is a floating caisson, 75m wide, 500m long and a 10m draught. The total water depth is 250m and the uniform current is 1 knot (0.5144 m/s) following the AB section direction (section AB is shown in Fig. 3). The non-hydrostatic TELEMAC-3D model mesh size is 5m with 11 planes, and the 7<sup>th</sup> plane is fixed at 11m below the surface using sigma planes above and below. Having the 7<sup>th</sup> plane fixed at -11m guarantees having 6 planes capturing the flow underneath the floating body. Tests were also done for 21 planes and a longer duration of 12h. Fig. 1 shows the surface longitudinal velocity as a percentage of the ambient current from the TELEMAC-3D results and Fig. 2 from the OpenFOAM and CFX results. It shows TELEMAC-3D results are very similar between the different combinations and therefore 11 planes and 5h simulation is sufficient in this case. Fig. 3 shows a plan view of the TELEMAC-3D velocity and Fig. 4 shows a cross section of the TELEMAC-3D surface velocity along the section AB of the caisson.

The TELEMAC-3D speed downstream of the caisson agrees well with OpenFOAM and CFX results. These conventional CFD models can include non-free surface flows and also can model free surface flow using volume of fluid (VoF) techniques. However, the reduction in the speed upstream of the caisson is greater in TELEMAC than the other two models. The caisson test case confirms that TELEMAC-3D can reproduce flows around a floating caisson, which is very similar in shape to a large ship such as a Floating Liquified Natural Gas (FLNG) vessel.

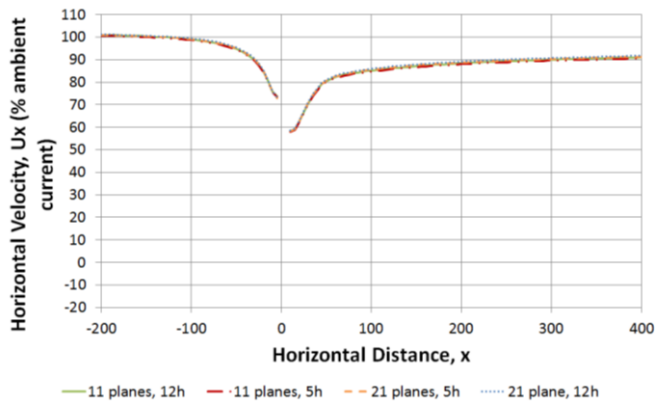


Figure 1: TELEMAC-3D longitudinal current velocity as a percentage of the ambient current at the free-surface along the section AB of the caisson for 90° current

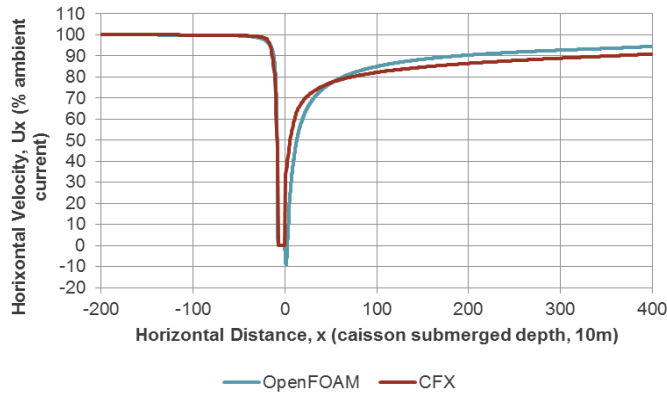


Figure 2: OpenFOAM and CFX longitudinal current velocity as a percentage of the ambient current at the free-surface along the section AB of the caisson for 90° current

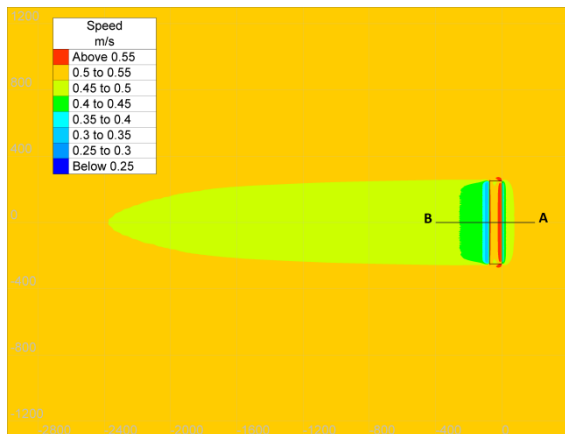


Figure 3: TELEMAC-3D velocity plan view

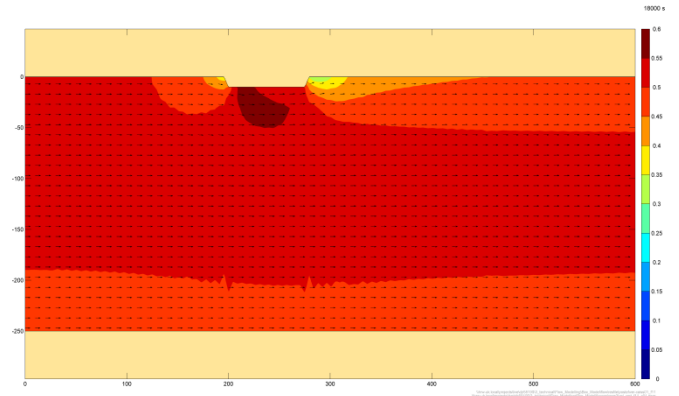


Figure 4: TELEMAC-3D velocity cross-section along the section AB of the caisson

### B. Live communication between TELEMAC and HR Wallingford ship simulator

HR Wallingford has two ship simulation centres, one in the United Kingdom and one in Western Australia. These simulators use time varying flow fields which are used to calculate the forces acting on simulated vessels. TELEMAC flow models are often used to provide these time varying flow fields and have been used for many years. Currently, these flow fields are an input to the ship simulator, i.e. the TELEMAC run is carried out beforehand.

As demonstrated for the caisson test case, large floating objects can have significant effects on the flow fields. In terms of navigation, this can have a significant impact on nearby vessels and is particularly important for vessel manoeuvring for side by side mooring such as LNG vessels approaching FLNG facilities.

In these situations the simulated flow field in the ship simulator must include the effect on the flow field of the FLNG and also respond to the movement (heading change) of the FLNG. For this reason, the TELEMAC-3D finite element flow model was developed to allow real time integration with the navigation ship simulator. The position and orientation of the ships is controlled by the ship simulator. This information is passed on to the TELEMAC-3D model, in which the position of the ship hull is imposed as a pressure field. The resulting current fields and water elevation are fed back from the TELEMAC model to the ship simulator in real time.

When first implemented, instabilities in the flow field occurred. They mostly depend on the size, shape and speed of the ship. Other factors are the water depth, the ambient flow and numerical parameters such as the time step, the number of planes and their distribution over the water column. In order to reduce instabilities, two approaches have been tried:

- The ship hull shape has been smoothed spatially to make the change in free surface less abrupt
- Time relaxation of the pressure fields: the pressure imposed depends on the requested value and the value at the previous time step. This prevents the free surface at a particular point changing too suddenly from one time to the next. This method is the same as that used

to compute culvert flows in TELEMAC while reducing unwanted oscillations.

- The mesh orientation and resolution was also optimised

Within the TELEMAC-3D model, these approaches combined significantly reduced model instabilities. However, when using the real time integration with the ship simulator, the ship hull cannot be excessively smoothed because it has an impact on the flow field used by the ship simulator. Overall, when setting a real case navigation simulation of a FLNG and vessels around it, the following parameters were used:

- The domain is a 4km wide and 6km long rectangle, with a uniform 240m water depth
- The TELEMAC-3D model mesh size is 6m near the FLNG growing to 100m on the boundaries (Fig. 5)
- 6 planes, spaced closer together near the surface than near the bed. More than 6 planes resulted in the TELEMAC-3D model running slower than real time
- Time step of 0.5s and time relaxation with a coefficient of 0.01. The corresponding formula (3) is given below, where  $P_t$  is the pressure at the time  $t$ ,  $P_{t-1}$  is the pressure at the previous time step and  $C_r$  is the relaxation coefficient

$$P_t = (1 - C_r)P_{t-1} + C_r P_t \quad (3)$$

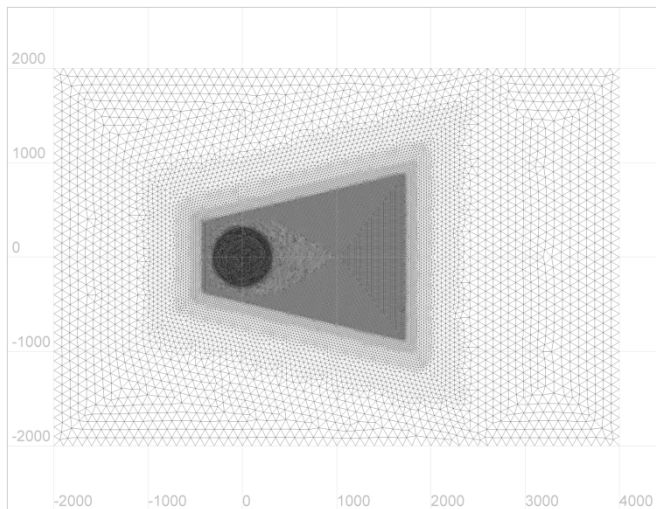


Figure 5: TELEMAC-3D mesh for a FLNG

The speed and velocity vectors, averaged over the top 20 m are plotted in Fig. 6, for an example with the FLNG stationary at 0°. The corresponding figure for the FLNG at 45° is shown in Fig. 7. Fig. 8 gives the vertical section through the middle of the FLNG, showing speed and velocity vectors for the 0° case.

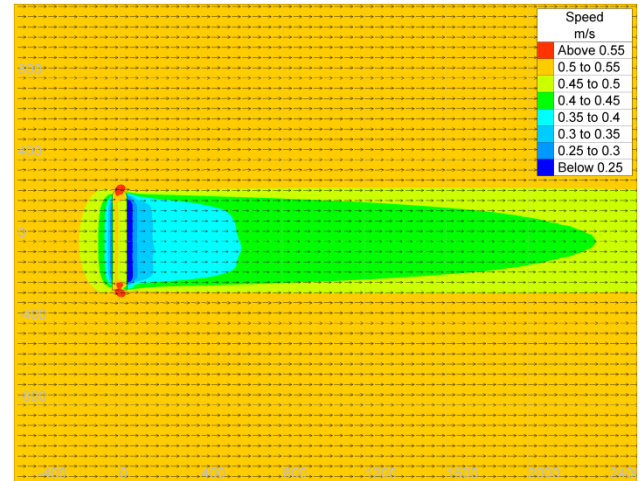


Figure 6: TELEMAC-3D Current sheltering – FLNG heading 0°

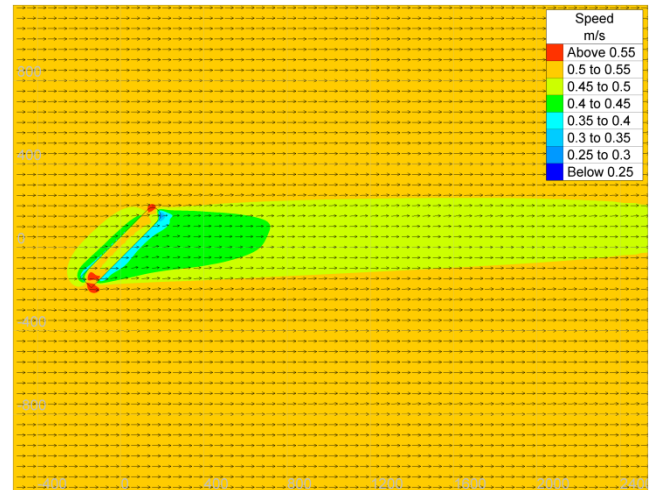


Figure 7: TELEMAC-3D Current sheltering – FLNG heading 45°

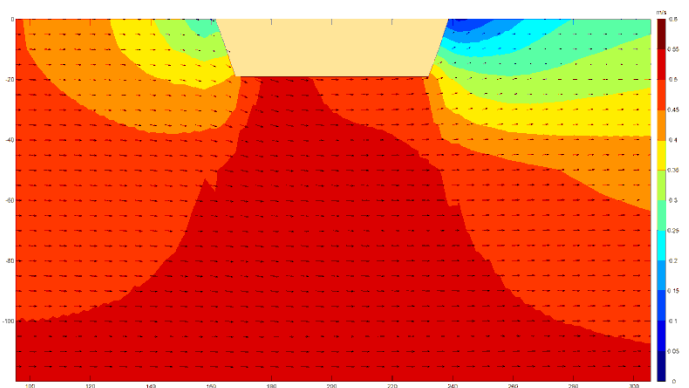


Figure 8: TELEMAC-3D midships vertical cross section – FLNG heading 0°

The real time integration of TELEMAC with the ship simulator was used to test manoeuvres as well as the familiarisation of tug masters and pilots for side by side FLNG operations (Fig. 9). Feedback was positive.



Figure 9: HR Wallingford navigation simulator snapshot – FLNG

### III. INFLUENCE OF TUG WASH

Some pilots have reported unusual flow when transiting through the channel of a harbour and passing another vessel near or at berth while being assisted by one or more tugs. These pilots believe that, under certain tide conditions, the thrust from the tugs builds up a localised flow condition across the navigation route with a potentially adverse effect on passing ships.

A tug wash model was developed and implemented in the existing harbour 3D flow model. The impact of the tug wash effects on the navigability of passing ships has been investigated using the HR Wallingford Ship Simulation System.

#### A. The tug wash model setup

The tug wash model was first setup and calibrated to estimate the likely tug thrust that may be present as computed by the TELMAC-3D flow model.

For this study a 27m ASD tug was used (Fig. 10). The corresponding characteristics are: 3,730kW 2x Schottel SRP1215; max rpm 314; diameter 2.4m; four blades; fixed pitch; two pods at 21.1m aft of bow fender, +/-2.3m from centreline, 3.7m below the waterline.

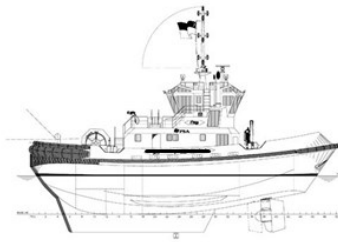


Figure 10: Tug Outboard Profile

In order to calculate the velocity in a thruster jet, the formulae (1) given by Blau and Van de Kaa [1] was used. The computed velocity for the tug, for 75% and 50% of its ballard pull is given in TABLE 1.

$$V_0 = C_3 \left( \frac{f_p P_d}{\rho_w D_p^2} \right)^{1/3} \quad (1)$$

- $P_d$  is the installed engine power (W)

- $f_p$  is the percentage of installed engine power
- $C_3$  is a coefficient equal to 1.17 for ducted propellers (determined by experiment)
- $\rho_w$  is the water density ( $\text{kg/m}^3$ )
- $D_p$  is the propeller diameter (m)

TABLE 1: THRUSTER JET VELOCITIES

$P_d$	$F_p$	$D_p$	$\rho_w$	$C_3$	$V_0$
$3730 \cdot 10^3$	75	2.4	1020	1.17	9.0
$3730 \cdot 10^3$	50	2.4	1020	1.17	7.8

The velocity computed is then implemented as a source of momentum within the 3D hydrodynamic model, at the appropriate location a few metres below the water surface. No water source is added, the TRISOU.f subroutine is modified to only add momentum.

#### B. The tug wash model validation

In order to validate the computed velocity for each tug thruster jet within the 3D hydrodynamic model, a schematic 3D model was used. The schematic model is a square 100m x 100m with a flat bathymetry at 2 metres below the still water level. A constant velocity in the tug thruster jets of 10 m/s for a period of 30 minutes was imposed, with a diameter of 1 m. Three vertical planes were used (one defining the surface, one the bed and the other in the middle, separating the water column in 2 layers, each 1 m thick) and the propeller was located at 1 m below the still water level. Horizontally, the mesh of the schematic model is based on a resolution of 1 m edges. Fig. 11 shows the current speeds at 45s for the full extent of the model. Fig. 12 and Fig. 13 show a close up view of the current speeds at 45s and 1800s, respectively.

Current speed time series were extracted at several distances from the propeller (white triangles in Fig. 12) and plotted in Fig. 14. All velocities shown in Fig. 11 to Fig. 14 were extracted from the middle plane, which is at the propeller level: 1 m below still water level.

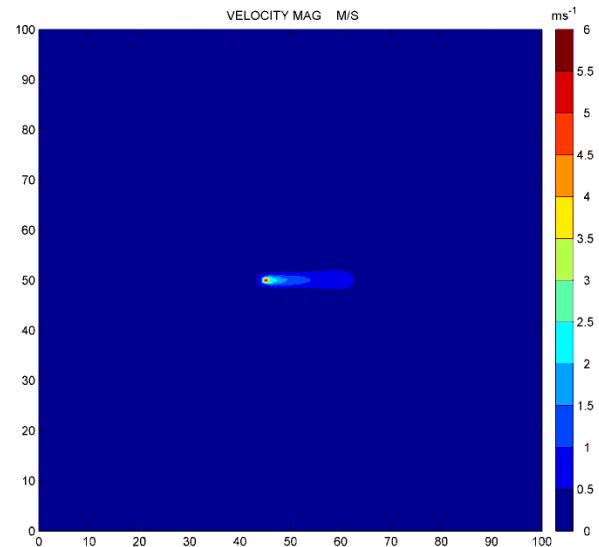


Figure 11: Schematic model full extent – Current speed at 45s



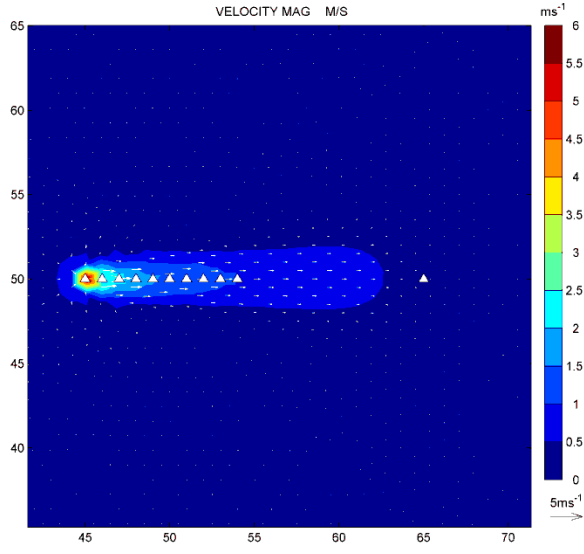


Figure 12: Schematic model – Current speed at 45s

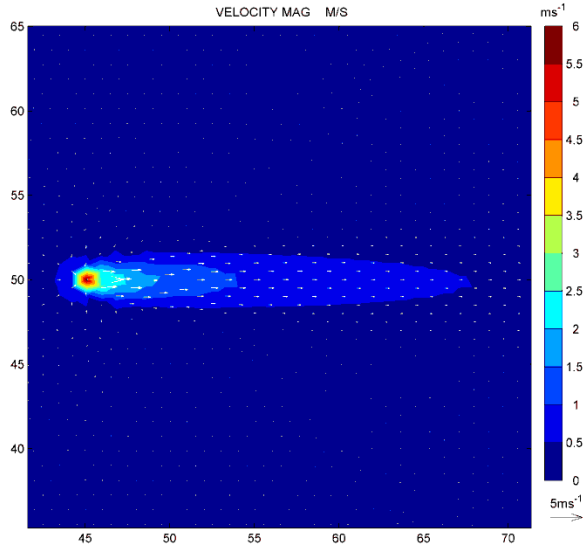


Figure 13: Schematic model – Current speed at 1800s

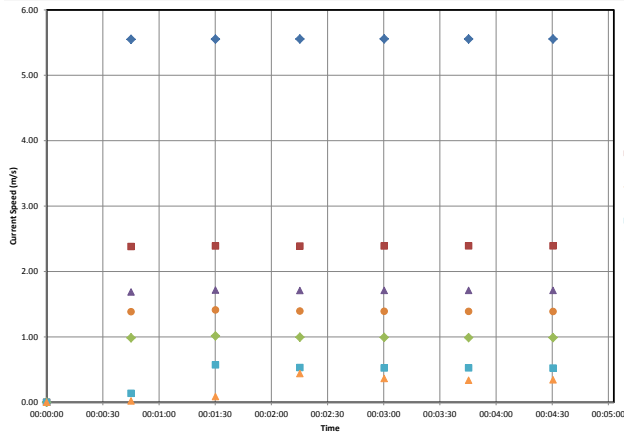


Figure 14: Current speed time series at several distances from the propeller, along the jet axis (White triangles in Figure 12)

In this schematic model, the current speed and its associated wake take about 3 minutes to achieve steady state,

as shown above in Fig. 14. The current speed also decreases quickly with the distance from the propeller along the jet axis. Fig. 15 gives a comparison between the model velocity decay and the formulation (2) of Albertson et al [2], also given in the PIANC guidelines ([3] Equation 8-1)

$$V_{axis} = \frac{1}{2C} V_0 \left( \frac{D}{x} \right). \quad (2)$$

- $V_{axis}$  is the flow velocity in the axis of the jet (m/s)
- $x$  is the horizontal distance from the propeller (m)
- $V_0$  is the efflux velocity (m/s)
- $C$  is a coefficient equal to 1.17 for ducted propellers (determined by experiment)
- $D$  is the diameter of the free jet (m)

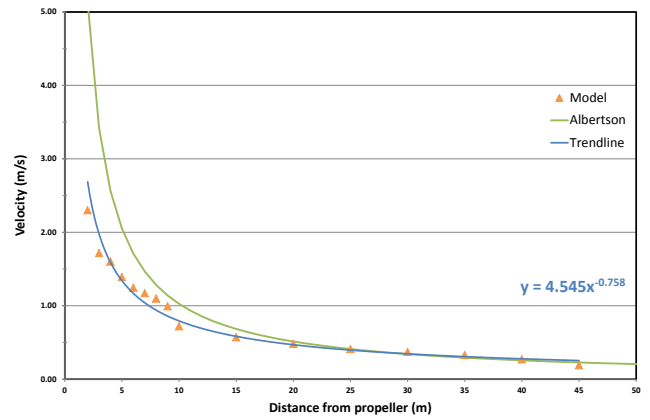


Figure 15: Velocity at 1800s from several distances from the propeller, along the jet axis

The trend line (the corresponding equation of which is shown in blue) and the formulation of Albertson are similar. The main differences appear to be within a 20 m distance from the propeller, that is, less than 10 propeller diameters downstream, where the jet has not yet reached its fully developed phase and therefore the estimation of velocities within the jet has a higher level of uncertainty. Note also that the formulation of Albertson assumes an infinite jet velocity at the propeller plane.

In the figures above (Fig. 11 to Fig. 15), the maximum velocity at the propeller is about 5.5 m/s. This is due to the fact that TELEMAC-3D applies the momentum within a volume, the size of which depends on the mesh resolution. Therefore, to be able to observe the propeller velocity 10 m/s applied in the model results, a finer mesh resolving the propeller dimensions would be needed. Separate validation tests were indeed carried out with finer meshes down to 0.5 m for the horizontal resolution and 0.25 m for the vertical resolution and an arbitrary propeller of 1 m in diameter, confirming the 10 m/s at the propeller location. This confirms that the effects of the propeller on the hydrodynamics are correctly represented. However, finer meshes also introduce numerical diffusion, resulting in lower velocities for distances greater than 20 m from the propeller (by about 0.1 m/s lower at 50 m). It is, therefore, not recommended to refine the mesh too much.

As the purpose of this study is to investigate flow patterns created a tens of metres away from the propellers, the comparison between the tug wash model and the Albertson formula is deemed acceptable and the 3D model is considered to be validated for this purpose.

### C. The tug wash model sensitivity

Sensitivity analyses specifically related to the tug jets were carried out, including the number of vertical layers representing the water column and the possible turbulence schemes. Also, in order to obtain a repeatable stable simulation so as to allow comparison between scenarios, sensitivity tests were performed to a number of numerical parameters such as model time step.

In the end, the parameters having the most influence on the results are summarised below.

- The horizontal and vertical turbulence scheme: Smagorinsky scheme (Smagorinsky 1963 [4]), used mainly for highly non-linear flows, with large scale eddy phenomena.
- The number of vertical planes: 12 was deemed to be the optimal number for computational efficiency.
- The time step: 2 seconds

### D. Passing ship results

A test case of a ship passing a recently moored vessel still under tug assistance was considered. The test case considered a scenario not long after slack water low tide (or the turning of the tide from ebb to flood) with three tugs in attendance holding a vessel at the berth which had been operating for 30 minutes at 75% power, then only two tugs operating at 75% for 30 minutes, which corresponds to a 9.0 m/s velocity. The aim of the test case was to determine if the tug wash had any influence on the passing ship.

The model bathymetry and the modelled water level time series around slack water low tide are presented in Fig. 16 and Fig. 17, respectively. The current speed and direction are plotted in Fig. 18. Time series were extracted at the location marked by a yellow triangle on Fig. 16. The current speed and direction contour plots are shown in Fig. 19 to Fig. 21 for snapshots. The colour scale only shows velocities up to 1m/s in order to see velocity patterns a couple of hundred metres away from the tugs. The velocity at the propeller is as described in Table 2 above, and diffuses quickly with the distance as shown in Fig. 15.

The results when integrated into real time ship simulation scenarios had a measurable effect on the passing ship with indications in the results of a recirculation eddy forming after sustained use of the tugs in this way.

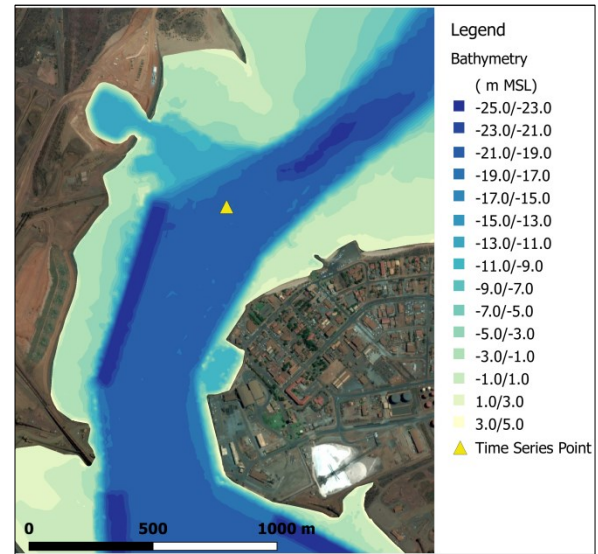


Figure 16: Bathymetry

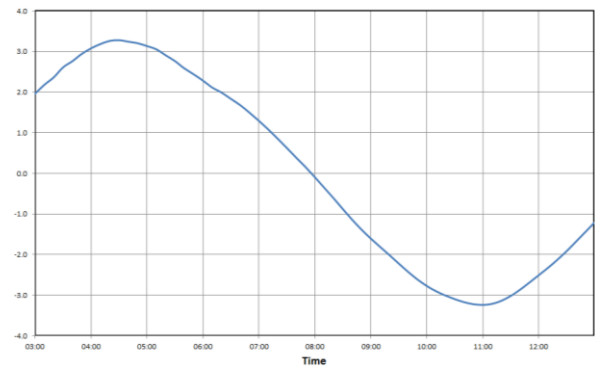


Figure 17: Free surface (m MSL) on the 30/01/2017

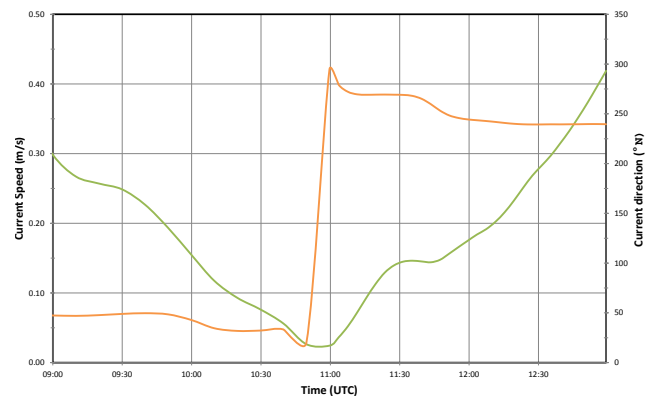


Figure 18: Current speed and direction on the 30/01/2017

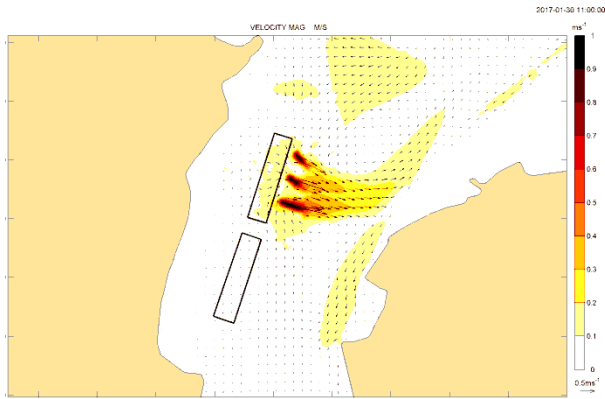


Figure 19: Current speeds and directions on 30/01/2017 at 11:00

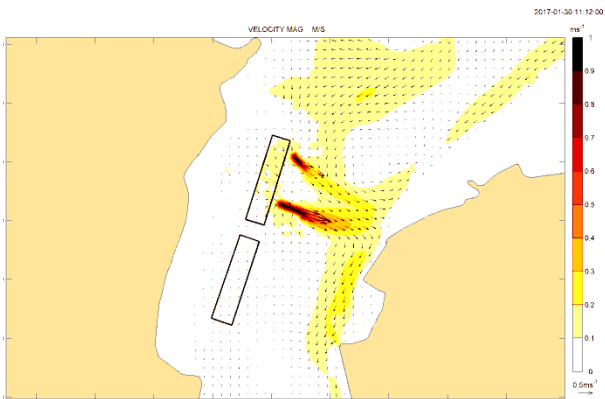


Figure 20: Current speeds and directions on 30/01/2017 at 11:12

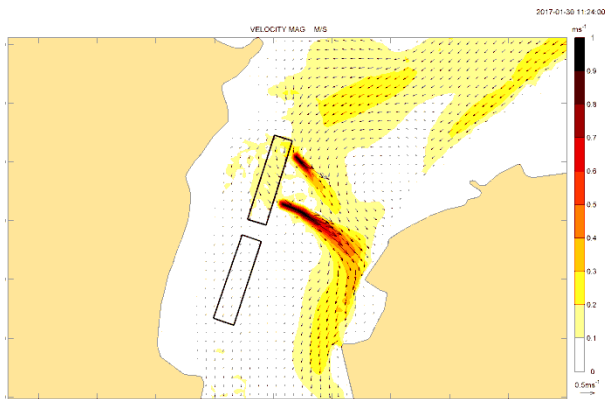


Figure 21: Current speeds and directions on 30/01/2017 at 11:20

#### IV. CONCLUSIONS

It has been found in practice that it is possible to use TELEMAC to make useful computations of the flow under and around floating objects. It is necessary, however, to treat such results with care for the following reasons:

- It is not clear whether the flow underneath the hull of the vessel is accurately modelled by imposing a pressure field. The shape of the hull is not guaranteed in this method to remain accurate as waves may travel across it. Also, the flow underneath the hull, especially if it is wide, is not a free surface flow in reality.

- The situation is more difficult if it is wanted to model a floating, moving vessel. The possibility of surface waves crossing the hull is now more likely resulting in the wrong hull shape. That is why a smooth representation of the hull shape was used. With a fine mesh in the horizontal it is possible to move the hull shape, as an air pressure variation, across the water surface. It is necessary to have several model cells to represent the transition from the deeper part of the hull to the water surface. The model time step needs then to be short enough to allow the vessel to cross each cell in several time steps. It is not expected that any rapidly moving vessel can be modelled without further refining the meshing processes. Another approach that may be useful for a moving vessel is to consider it instead to be stationary in a moving water flow.

It is clear that using a free surface flow model like TELEMAC to model a vessel's hull moving across the model mesh is an extension beyond normal use of such a flow model but once verified (e.g. against CFD) can by provide computationally efficient and representative flow fields resulting from the effect of floating structures.

It has been found that tug wash (propeller jet) can be usefully modelled in TELEMAC-3D for examples where the tug wash can have an influence on ships that are impacted by it.

#### REFERENCES

- [1] Blaauw, H.G. & E.J. van de Kaa (1978) *Erosion of bottom and sloping banks caused by the 2775 screw race of manoeuvring ships*. WLDelft Publication no. 202, Proc. 7th Intern. Harbour 2776 Congress, Antwerp, May 1978.
- [2] Albertson, M.L., Dai, Y.B., Jensen, R.A., Hunter Rouse (1950), *Diffusion of submerged jets*, 2765 ASCE Transactions Paper no. 2409, New York, 639-664.
- [3] PIANC (2012), Guidelines for berthing structures, related to thrusters.
- [4] SMAGORINSKY, 1963. General Circulation Experiments with the Primitive Equations, Monthly Weather Review, vol. 91, issue 3, p. 99.